Seismic Modelling of an AGR Nuclear Reactor Core

Authors:
Bruce Duncan, Atkins
Berislav Kralj, Atkins

Correspondence:
Bruce Duncan
Atkins
240 Aztec West
Park Avenue
Almondsbury
Bristol
BS32 4SY
United Kingdom
Phone: +44 1454 288 756
Fax: +44 1454 616 480
Email: bruce.duncan@atkinsglobal.com

ABSTRACT

The work concerns the seismic response of an Advanced Gas-cooled Reactor (AGR) core. An LS-DYNA finite element model has been developed that represents each graphite brick. The contact between the bricks and their keys has been modelled with non-linear coaxial spring/dampers allowing for limited free motion. The model is a three dimensional representation of all the core layers and a neutron shield, making use of a symmetry by only modelling half of each layer. Different layers and groups of bricks have been allocated different spring/damper properties in order to accommodate differing initial geometry and levels of irradiation flux exposure. The core's restraint system has also been modelled. Bespoke programs have been developed in Visual Basic for Applications to automatically generate the key file used by LS-DNYA and deal with the post-processing of results files. Sensitivity studies were done to assess the effects of changing the magnitude of the event and the effects of irradiation on the bricks.

Keywords

Nuclear, AGR, Reactor, Power Generation, Seismic analysis, Earthquake, LS-DYNA, Finite Element, Visual Basic, Excel, Graphite, Irradiation

INTRODUCTION

With the risk from earthquakes being low in Britain, the design of earlier AGR (Advanced Gas-cooled Reactor) nuclear power stations designed during the late 1960's, did not consider the consequences of seismic loads. More recently, however, it was accepted that the stations could (even though with a very low probability) experience a seismic event of significant severity. Consequently, it became a requirement to demonstrate that these earlier nuclear power stations, of which Hinkley Point B and Hunterston B were the first to start operating, can be safely shut down in the case of a seismic event. In order to satisfy this requirement, British Energy asked Atkins to develop a 3-dimensional computer model capable of predicting the seismic responses of typical AGR reactor cores and of assessing their behaviour during an earthquake event, with particular emphasis on evaluating:

- Loads on individual keys
- Loads on the elements of the restraint structure
- Patterns of peak brick displacement through each layer
- Double crack separation/shear
- Fuel stringer squeezing force
- Retention of key engagement
- Control rod insertion path alignment during and following the event

CORE MODEL

Core description

The AGR core, measuring 10m x 11m, comprises of stacks of graphite bricks providing vertical channels for fuel assemblies, control rods and coolant flow. The core is supported by plates resting on a diagrid that is supported on a ring of columns. Lateral stability of the core is provided by a restraint structure consisting of restraint rods, restraint beams, centralising brackets and Warwick links tied into the restraint ring, which is attached to the surrounding boiler shield wall, Figure 1. The stacked bricks are interconnected on plan by a system of keys and keyways allowing the core to expand and contract thermally whilst retaining its cylindrical shape, Figure 2.
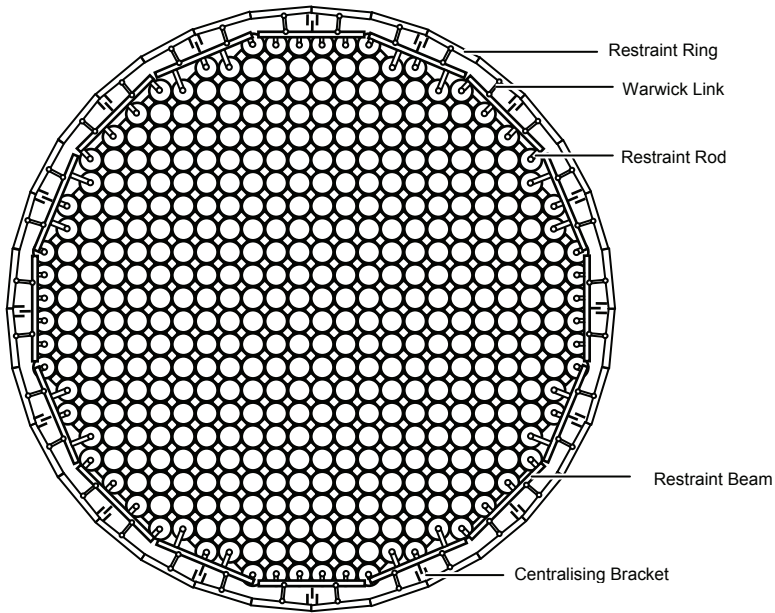
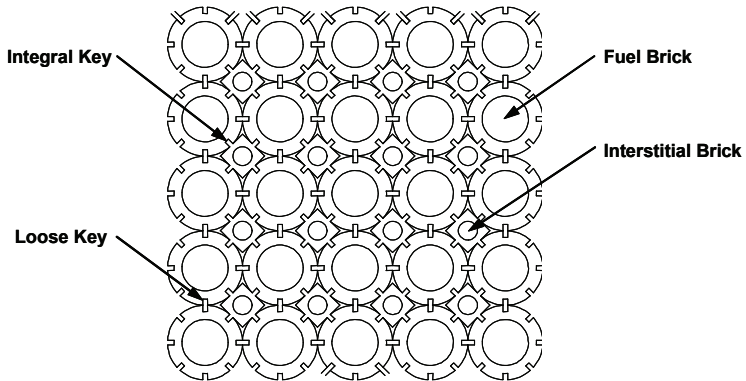FIGURE 1: Schematic representation of a graphite core layer

FIGURE 2: Schematic representation of the keying system on plan

The graphite bricks are also keyed together vertically by their end features, while offsets in the arrangement of fuel and interstitial bricks would prevent any gross relative shear displacements (Figure 3). The finite element model of the core has been

developed so that it simulates all of the significant interactions between adjacent bricks, and between the bricks and the core restraint structure.

FE Model - Bricks

The aim of the work is to investigate the dynamics of the core, but is not to consider stresses caused in the bricks in detail. Because of that, and in order to keep the size of the finite element model manageable, a decision was made to not adopt a 3D model of each brick as a solid body (with 'brick' type 3D finite elements) with contact surfaces. Instead, each brick in the core is modelled as a rigid, vertical body, with inertial and other properties attached to it. In order to represent possible interactions with the adjacent bricks along its height, each rigid body was attributed seven planes in which rigidly connected nodes were to be defined in order to attach spring/dampers to the brick, for modelling contact between adjacent bricks, Figure 4. The model made use of symmetry by only modelling half of each layer.
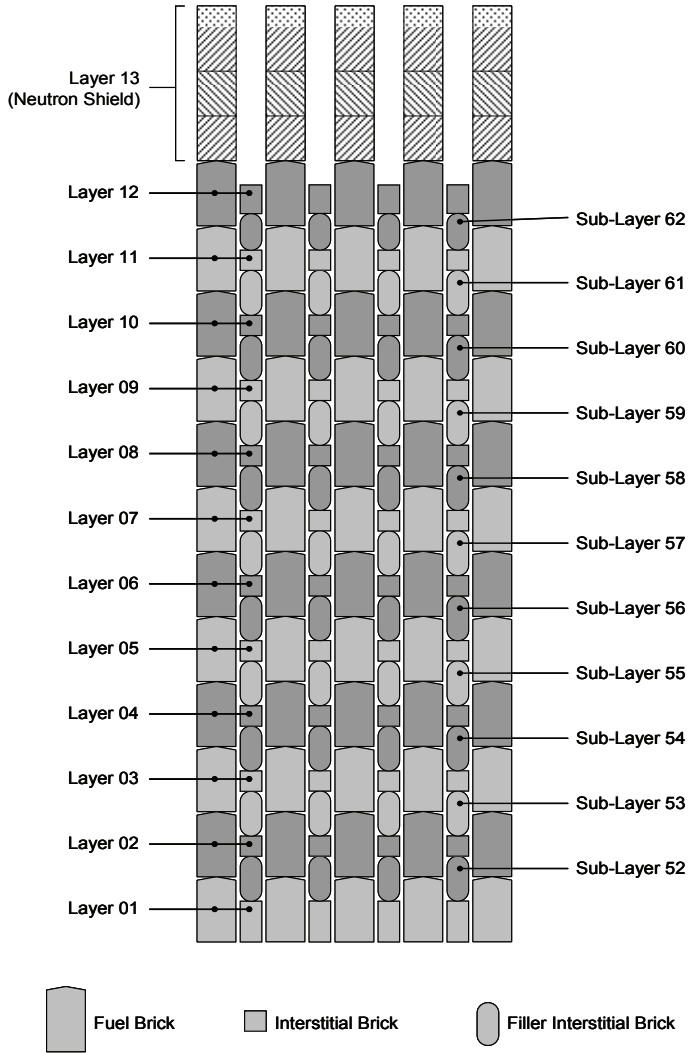
FIGURE 3: Schematic representation of a keying system in vertical plane

Brick interactions

Bricks can interact with adjacent bricks through a number of mechanisms as summarized in Table 1 below. Since the main aim of the work is to investigate dynamics of the core when subject to seismic loads, contacts between the bricks are

predominantly short duration impacts. Impact stiffness of the contact was based on values measured in experiments, while a coefficient of restitution of 0.65 was used. In order to simplify the model and allow more control over its behaviour, all of these interactions were modelled through a number of springs and dampers connecting relevant nodes.
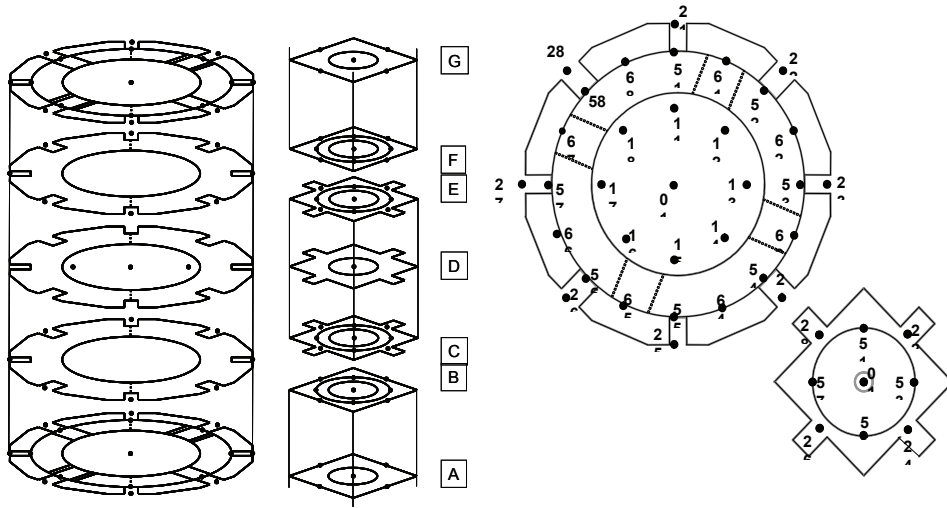
FIGURE 4: Schematic representation of a vertical discretisation of bricks (left) and all the possible nodes in a single plane
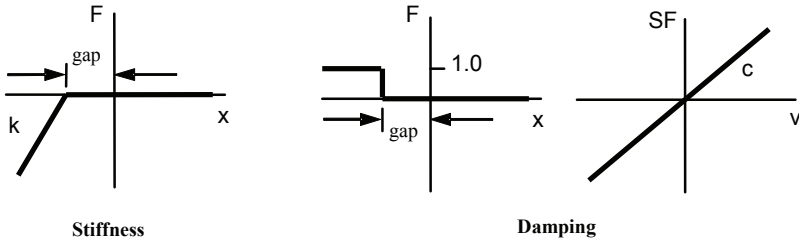
Nodal Plane	Vertical Position in Fuel Brick Column	Vertical Position in Interstitial Brick Column	Interactions
A	FB bottom surface	FIB centre of mass	FB – FB lateral direct contact FB – FIB lateral direct contact FB connection with FB below
B	N/A	FIB top surface	FB – FIB lateral direct contact FIB connection with IB above
C	Bottom of loose bearing key	IB bottom surface	FB – IB lateral direct contact FB – FB loose key FB – IB integral key IB connection with FIB below
D	FB centre of mass	IB centre of mass	None
E	Top of loose bearing key	IB top surface	FB – IB lateral direct contact FB – FB loose key FB – IB integral key IB connection with FIB above
F	N/A	FIB bottom surface	FB – FIB lateral direct contact FIB connection with IB below
G	FB top surface	N/A	FB – FB lateral direct contact FB – FIB lateral direct contact FB connection with FB above

Key: FB – Fuel Brick   IB – Interstitial Brick   FIB – Filler Interstitial Brick
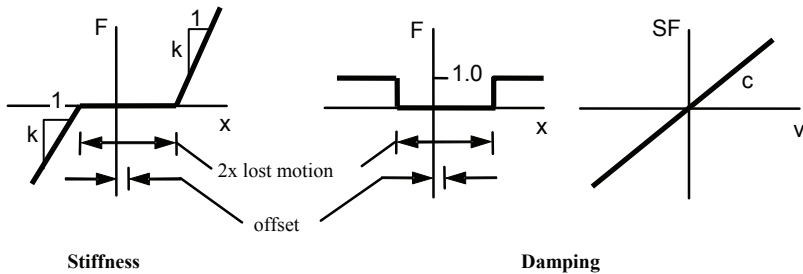
TABLE 1: Summary of interactions in the keying system and their locations

The keying system was designed to allow some free motion before becoming engaged by specifying clearances in the system. In order to model these clearances, the springs modelling the impact behaviour have force displacement curves as depicted in Figure 5. Energy dissipation during the impact, resulting in a restitution coefficient less than 1, is modelled as an equivalent viscous damping using dashpot elements in parallel with the springs. LS-DYNA modeling allows for this by defining the viscous damping force to be proportional to the velocity, with a proportionality ratio equal to the dashpot damping. Each damper is defined by both a displacement graph and a velocity graph, with their ordinates multiplied together, so that they only come into play whilst contact is actually made between the components.

Direct Brick Contact

Stiffness   Damping

Shear Key/Keyway Contact

Stiffness   Damping

Key
x = deflection   F   =   k = Stiffness (dF/dx)
v = velocity   SF = Scale Factor   c = Damping Coefficient (dF/dv)

FIGURE 5: Definition of contact spring and dashpot properties

Dimensions of the bricks and keys, as well as clearances in the keying system were calculated based on the geometry of the bricks after a prescribed number of full power years of operation, i.e. the effects of irradiation on dimensions, stiffness and shape of the bricks, keys and keyways were taken into account. The developed model has the capability to account for potential damage induced in bricks due to irradiation, thermal or other defects. For example, development of a single vertical through-the-wall crack would cause opening of the brick's cross section into a C shaped profile. This would lead to changes in the apparent radius of the brick, as well as offsetting and changes in the shear keyways' clearances. If a secondary crack develops, the brick would split vertically into two half-bricks. The developed model was designed to be able to simulate these. Positions of the cracked bricks and crack plane orientations can be randomly distributed within the core's more irradiated zones with prescribed probabilities of primary and secondary cracks occurring.

Connections of the bricks within a stack are twofold; horizontally, the keying system at the ends of the bricks only allows limited relative horizontal motion of the bricks; vertically, the ends of the bricks are designed so that the moderator brick above rocks on the brick below (i.e. there is no full area contact). This rocking feature was designed to help evenly distribute stresses due to thermal expansion. For the rocking motion, two dominant rocking directions were designed; higher stiffness for rocking in a core's tangential direction, and lower stiffness for rocking in the core's radial direction. These end features were also modelled by nonlinear springs connecting relevant nodes on brick ends, Figure 6.
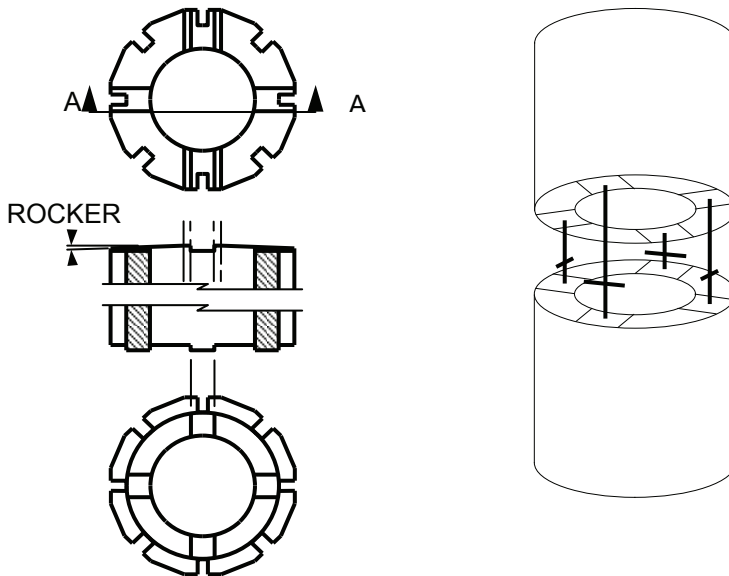

FIGURE 6: Modelling of axial (vertical) features

RESTRAINT STRUCTURE MODEL

Description
The main components of the restraint structure are depicted in Figure 1, and comprise of:

- Restraint links. One end of the restraint links is connected to peripheral bricks via a spigot at a mid-layer plane whilst the other end is fixed into a restraint beam.
- Restraint beams. There are 16 beams per horizontal mid-layer plane, each beam supporting four restraint links.
- Two Warwick links per beam, creating a trapezium on plan. The Warwick links are ball jointed at both ends.
- One centralizing bracket per restraint beam designed to locate the restraint beam circumferentially in relation to the restraint ring beam. The brackets were designed to allow free relative motions in radial and vertical directions.
- The Warwick links are connected into the restraint ring beams, which are connected into the boiler shield wall.

FE model
The FE model of the core models the restraint structure up to the boiler shield wall, which is assumed to be rigid and is where the input motion due to the seismic event is applied to the model. Figure 7 shows a finite element model of the restraint structure in more detail. Each restraint beam with its associated restraint links was modelled as a single rigid body. Flexibility of the components and clearances in the system were modelled using direct springs and dampers. The centralising bracket spring properties were developed by considering a separate, more detailed component finite element model incorporating material and geometrical non-linearity. Results from this component model were than transformed into a load-displacement curve for the respective spring.
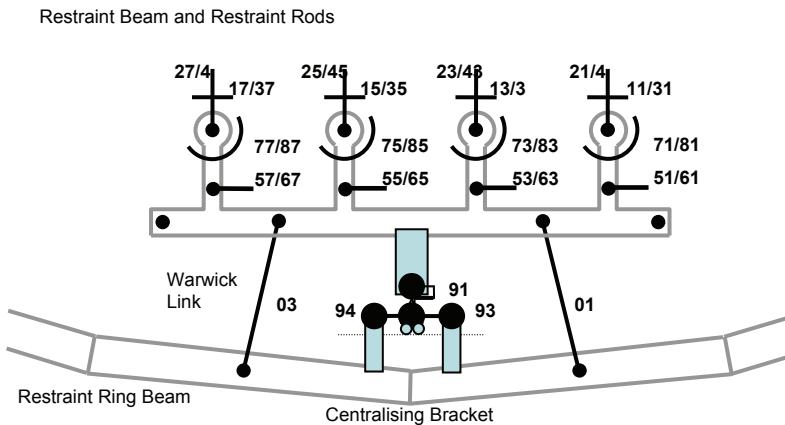
Restraint Beam and Restraint Rods

FIGURE 7: Diagram showing numbering of restraint structure elements

KEYFILE GENERATOR

It was recognised that general purpose finite element mesh generators are not suitable for developing this type of finite element model. Because of that, a decision was made to develop a purpose-built program capable of automatically generating a complete keyfile for inputting data to LS-DYNA. This unusual approach allowed the team to efficiently process a series of different simulations, enabling an assessment of the core's sensitivity to variables, e.g. earthquake intensity, number and distribution of cracks. The keyfile generator program was written in Visual Basic for Applications, in combination with Excel. While it is generic in certain aspects, it is applicable without modifications only to a single core design. The generator reads results of preliminary calculations defining the geometry of the core and the keying system, restraint structure properties, probability of damage to the core, and automatically creates an LS-DYNA keyfile. The total number of nodes used in the model was 283202, whilst the number of elements totalled 264296.

The topology, i.e. numbering system, of the FE model's constituents has been developed such that it is quickly apparent to an experienced user the exact whereabouts of any particular node or element just by looking at the number, and similarly any given component's number can quickly be ascertained by following the topology calculation's conventions.

POST-PROCESSOR

The main aim of the work is to demonstrate that, during and following the seismic event, the core can be safely shut down and held down. Based on this requirement, the following aspects of the core performance need to be assessed for the seismic analysis:

- Integrity of the keying system
- Integrity of the core support structure
- Fuel and control channel alignment, i.e. interaction of the channels with the fuel stringers and control / sensor rods to allow free movement of these components and free flow of coolant
- General level of displacement of graphite bricks

Again, the general purpose finite element post-processor is not suitable for dealing with models like this and displaying results in an appropriate format for assessment. Because of that, a bespoke post-processing tool was developed which automatically extracts all the required results from LS-DYNA output files, carries out the required analyses and assessment of the components, and displays results in a user friendly format.

SAMPLE RESULTS

The analyses are run for an acceleration time history extracted at relevant locations from the calculated responses of an earlier and simplified finite element model of the reactor. A full time history analysis was carried out with a time step set sufficiently short to prevent any instability in results. Sensitivity studies were carried out to investigate the effects of various time-step sizes on the model behaviour. At the beginning of the analyses, the model is left for a certain period of time to settle down into an equilibrium position, before the seismic accelerations are applied. The analysis is run for the whole duration of the assumed seismic event. Outputting results of the analyses at each time step would lead to a large results file, making them un-manageable. Because of that, the results output frequency is set to be variable during the analyses, with the highest frequency during the strong motion period of the seismic event. Sensitivity studies were carried out to assess the effects of different output sampling rates on the accuracy of the presented results.
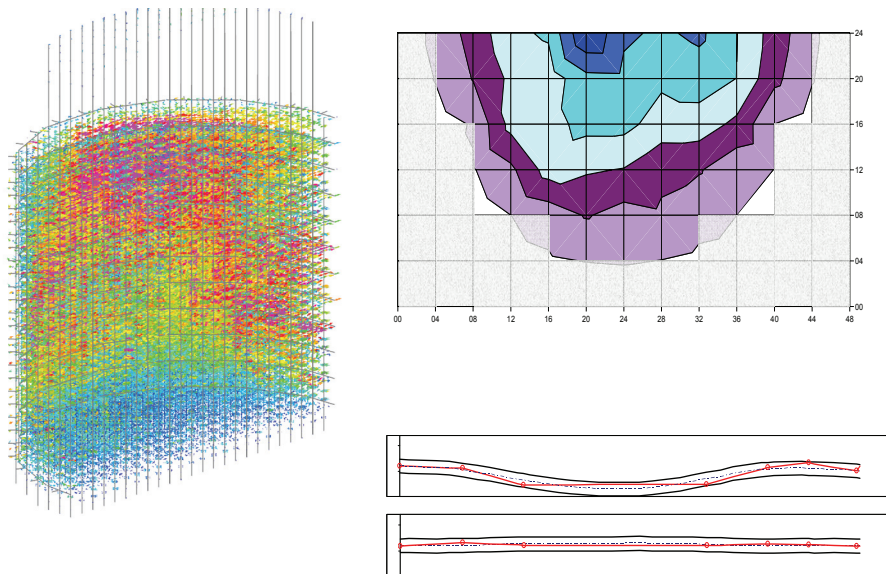
FIGURE 8: Example results; contour plot of velocities (left), displacement contour plot at one layer (top right), deformed channel profile with sensor rod (bottom right)

CONCLUSION

A finite element model was developed to evaluate the effects of seismic event on a reactor core at various times in life. Effects of potentially damaged bricks on the core response were modeled. Purpose-built programs were developed for generating a finite element mesh for LS-DYNA and post-processing results for analysis.